

CFD Workbench

# FreeCAD-CFD Workbench

Tutorial 4: External aerodynamics of a UAV

# CFD Workbench

## WORKBENCH

This workbench aims to help users set up and run CFD analysis. It guides the user in selecting the relevant physics, specifying the material properties, generating a mesh, assigning boundary conditions and setting the solver settings before running the simulation. Where possible best practices are included to improve the stability of the solvers.

## INSTALLATION

### WINDOWS:

- <https://www.freecadweb.org/wiki/Download>
- Install CfdOF from Tools | Addon manager
- Go to Edit | Preferences | CFD to check and install dependencies

### LINUX:

- [https://www.freecadweb.org/wiki/Install\\_on\\_Unix](https://www.freecadweb.org/wiki/Install_on_Unix)
- Install CfdOF from Tools | Addon manager
- Install OpenFOAM (5.0 recommended) (<https://openfoam.org/download/>)
- Install Paraview (tested with 5.0.1)
- Optional - Install GMSH (optional, 2.13+)
- Go to Edit | Preferences | CFD to check dependencies and install cfMesh

## LATEST INFORMATION

Please see the CfdOF [README file](#) for up-to-date information.

## LEAD DEVELOPERS

Johan Heyns (CSIR, 2016-2018) [jaheyns@gmail.com](mailto:jaheyns@gmail.com),

Oliver Oxtoby (CSIR, 2016-2018) [oliveroxtooby@gmail.com](mailto:oliveroxtooby@gmail.com),

Alfred Bogaers (CSIR, 2016-2018) [abogaers@csir.co.za](mailto:abogaers@csir.co.za),

# UAV aerodynamics

- To demonstrate how to model viscous flow over an unmanned aerial vehicle.
- Study the effect of including the camera gimbal



CAD obtained from:

<https://grabcad.com/library/uav-40>

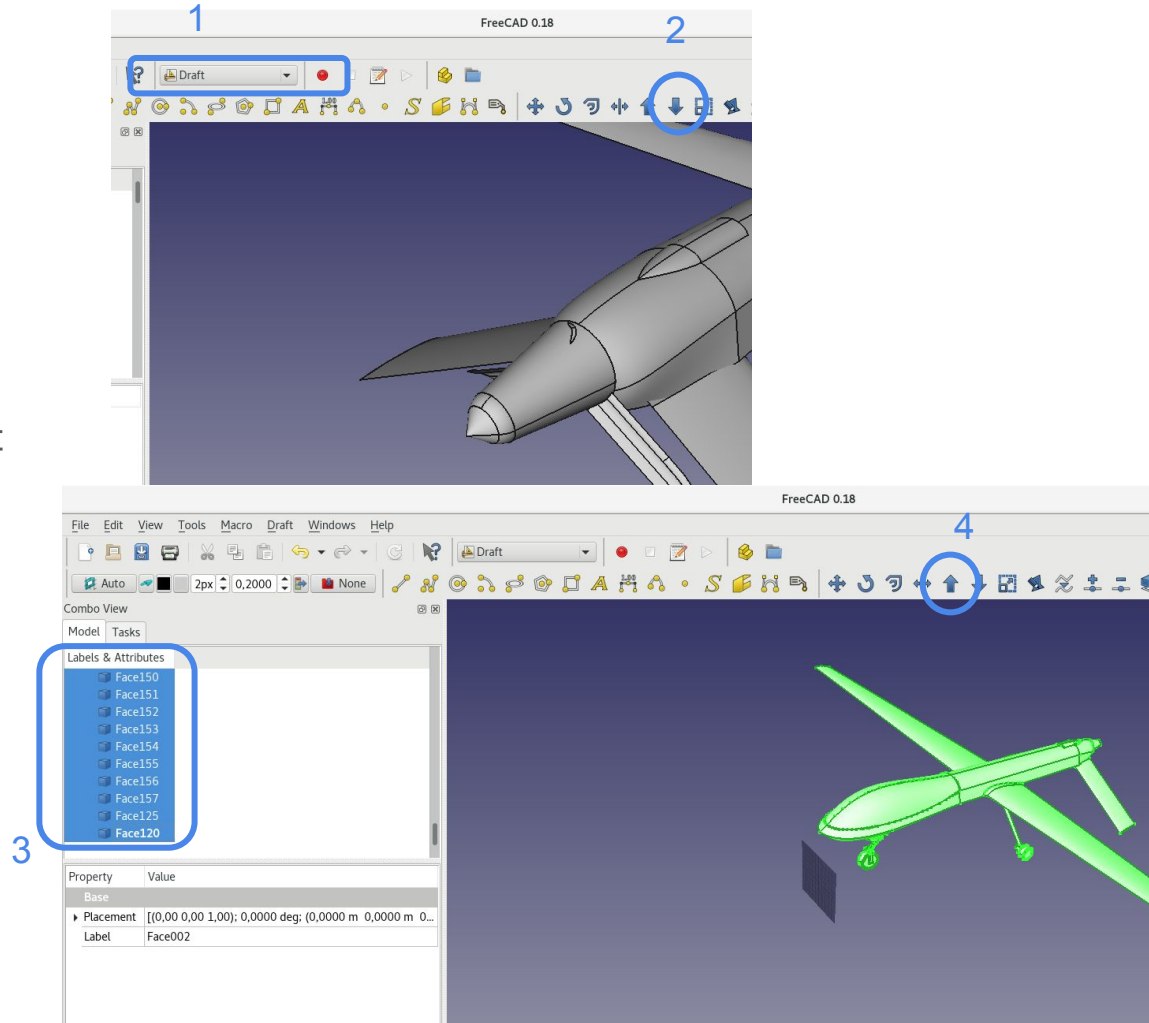
Created by Shadman Sakib

Non-commercial use permitted with attribution

# Part Design

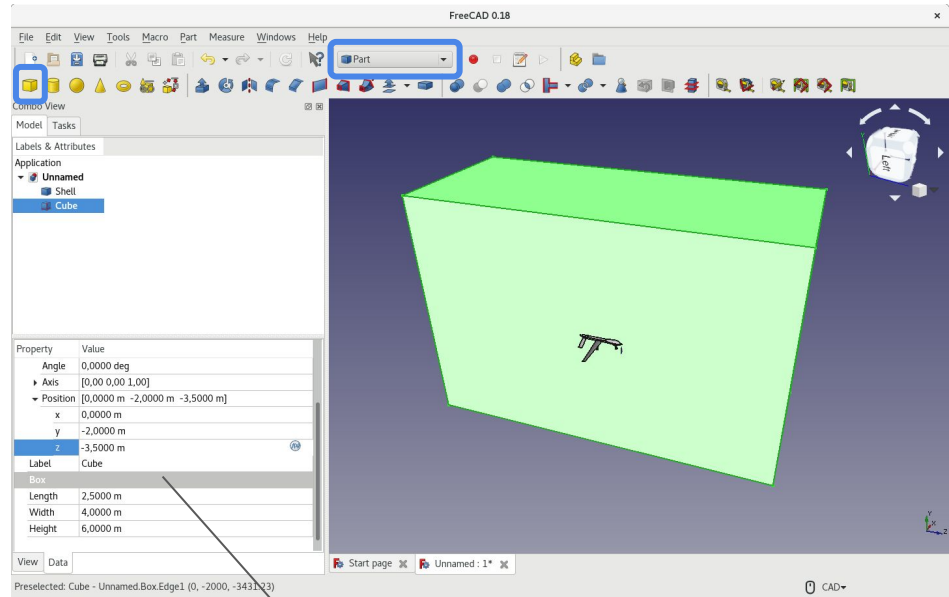
# Geometry

- Open the supplied .igs file in FreeCAD
- We wish to remove the propeller blades for the analysis.
- Open the 'Draft' workbench, select the 'UAV' object in the tree view, and click the 'Explode' button.
- Select and delete each face of the propeller.
- Select all faces and click the 'join' button to re-combine them.



# External flow domain

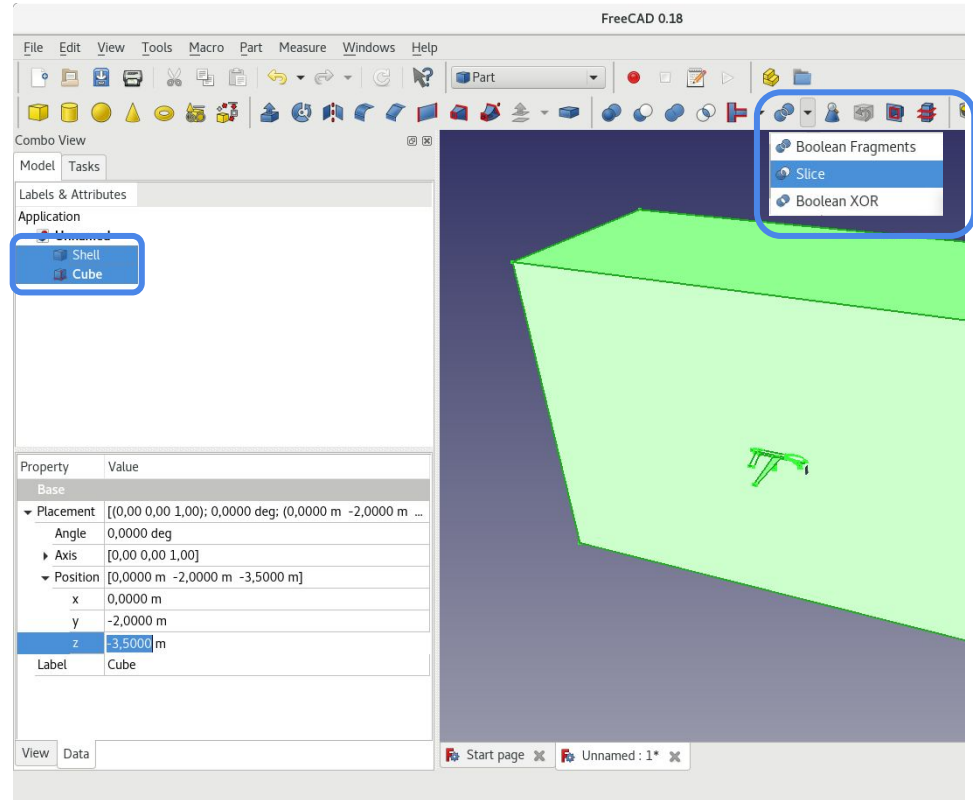
- Open the 'Part' workbench and create a cube to hold the external mesh.
- Set the position and dimensions as shown.
- For the final analysis, the far field domains should be much further from the body
  - Rule of thumb is 10 times its characteristic length
  - We choose closer boundaries for a quicker preliminary analysis
- We are cutting the body in half in order to save time by simulating only half of the symmetric domain



Axis	[0,00 0,00 1,00]
Position	[0,0000 m -2,0000 m -3,5000 m]
x	0,0000 m
y	-2,0000 m
z	-3,5000 m
Label	Cube
Box	
Length	2,5000 m
Width	4,0000 m
Height	6,0000 m

# External flow domain

- To cut out the aircraft from the outer domain, **first** select the 'Cube' object, then the 'Shell' object (holding Ctrl), and choose the 'Slice' operation in the Part workbench
  - The 'Slice' operation cuts away from the first selection using the second (and any further) selections
  - The 'Cut' operation is an alternative, but only works for watertight solid objects (often not the case for imported CAD).

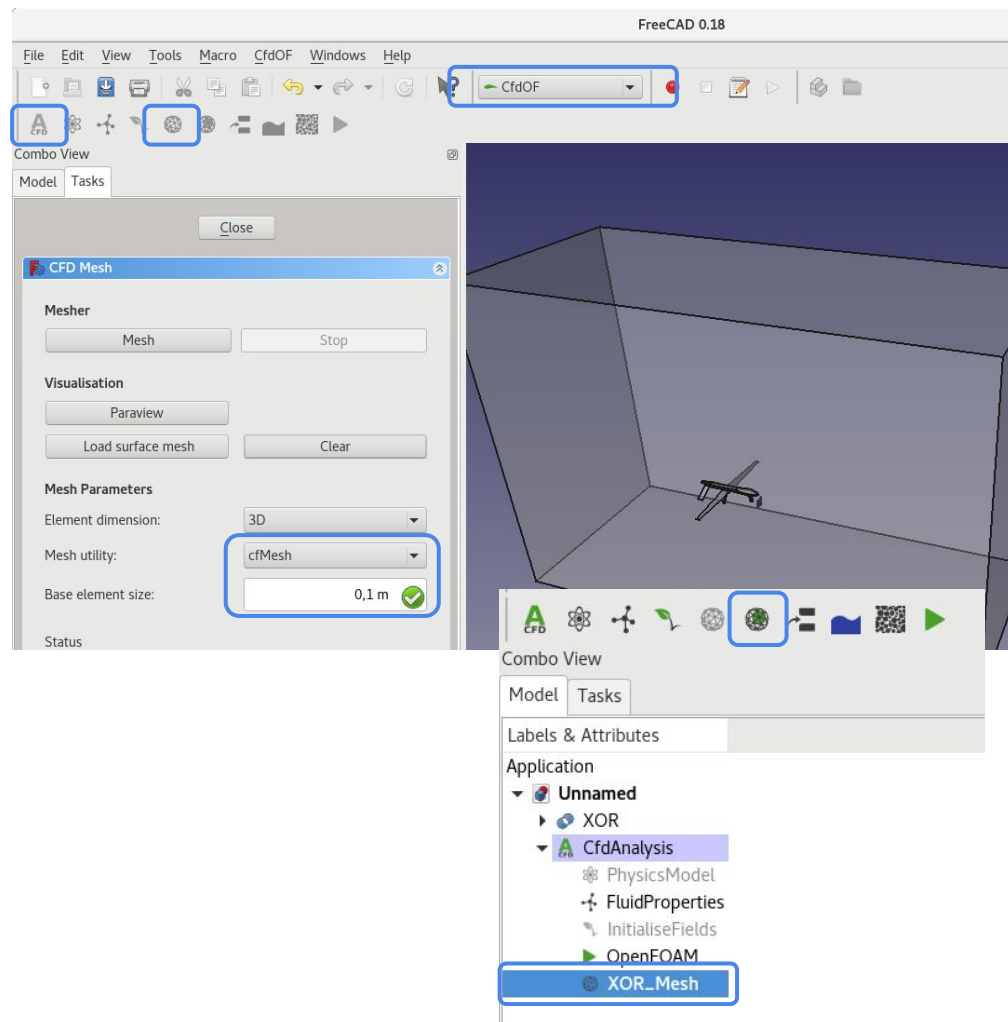


# Mesh generation and mesh refinement with cfMesh



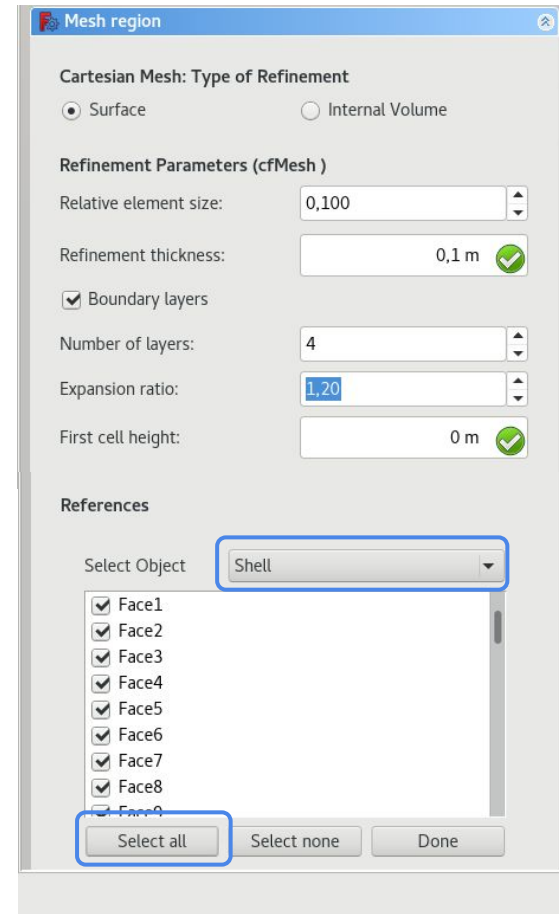
# Create Mesh object and refinement region

- Activate the CfdOF Workbench
- Create an 'Analysis' object
- Select the 'Slice' object and click the 'Mesh' button.
- Select the cfMesh mesher and a base element size of 0.1 m
- Select the mesh object and click the 'Mesh region' button



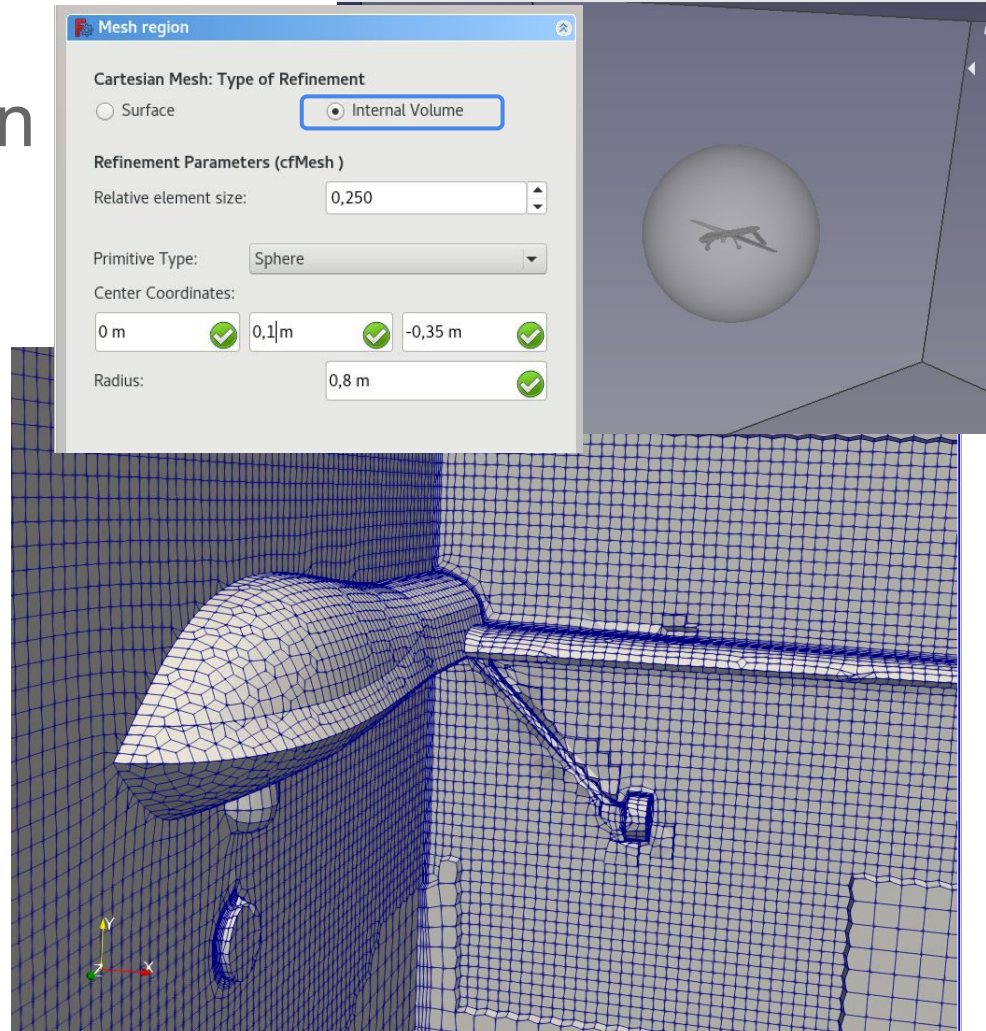
# Surface refinement region

- For 1/10th refinement within 0.1 m of the body, input a refinement thickness of 0.1m and a relative element size of 0.1.
- Here we choose 4 boundary layers with an expansion ratio of 1.2.
  - See Tutorial 3 for more information on boundary layers
  - For non-smooth geometries, the mesher and/or solver may struggle if too many boundary layers are added.
- To easily select the entire body of the aircraft, click 'Select from list', then choose the 'Shell' object and click 'Select all'



# Volume refinement region

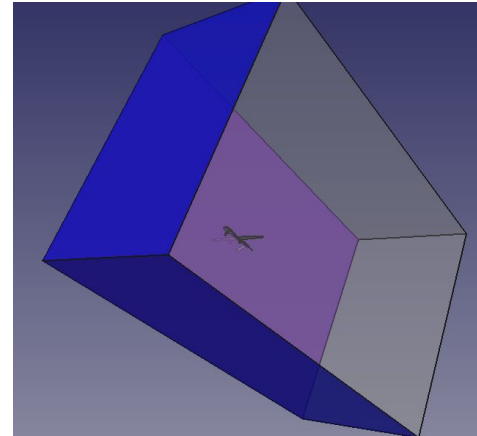
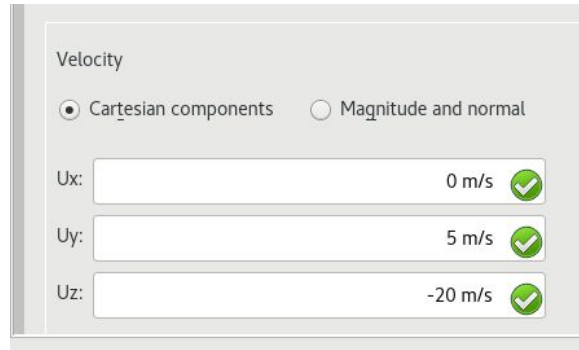
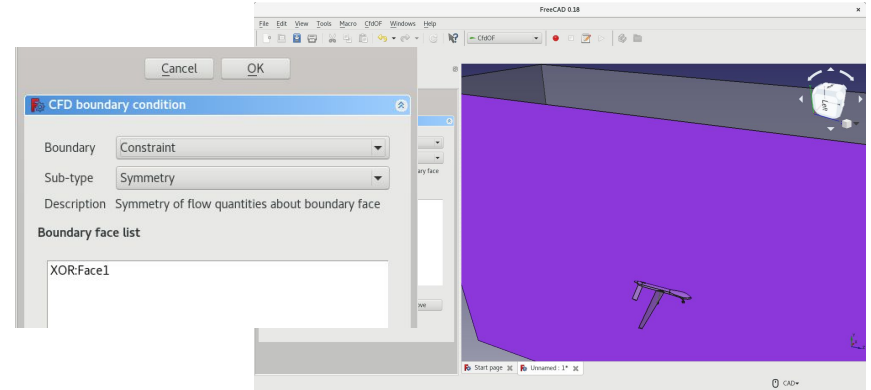
- To achieve a more gradual refinement from the far field to the surface of the body, we introduce an additional volume refinement
- Select the 'Slice\_mesh' object and click the 'Mesh region' button as before
- Select 'Internal volume' and enter the parameters as shown
  - cfMesh currently only supports spherical and rectangular refinement zones
- Return to the mesh object and click 'Mesh'.
- Click 'Paraview' to view the result.



CFD analysis

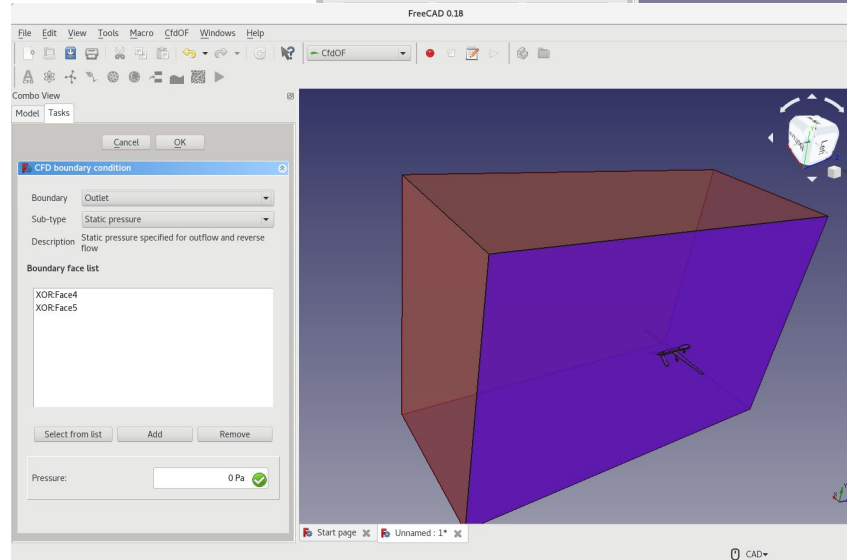
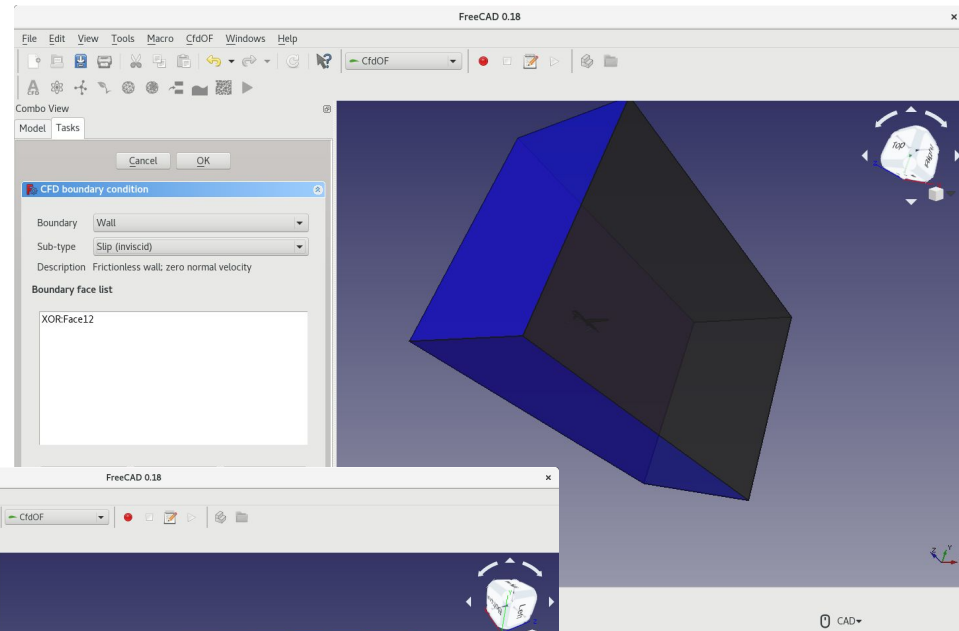
# Add boundary conditions

- Create a CFD boundary condition of 'Constraint' type 'Symmetry' for the central cutting face
- Create a 'Uniform velocity' inflow boundary condition with the parameters shown below and add the lower and front faces.
  - Flight at 74 km/h and 14° angle of attack



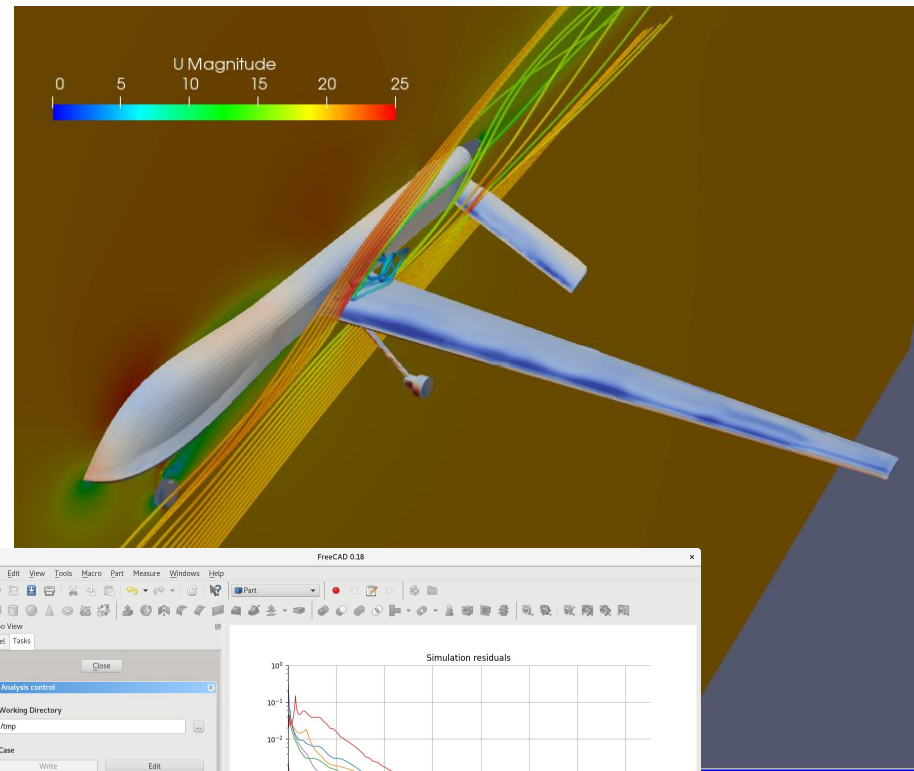
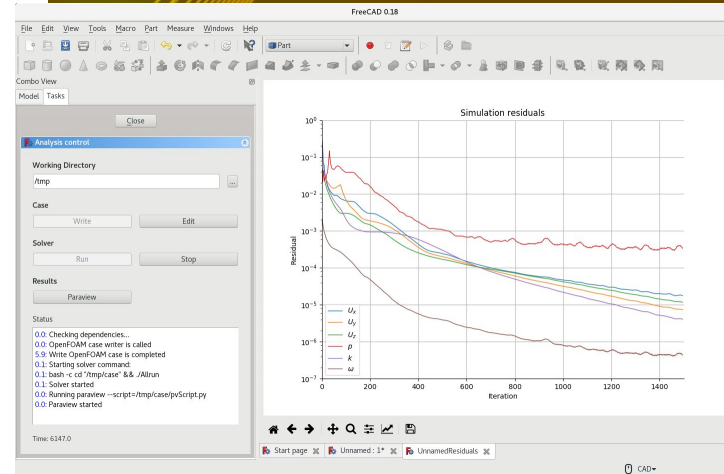
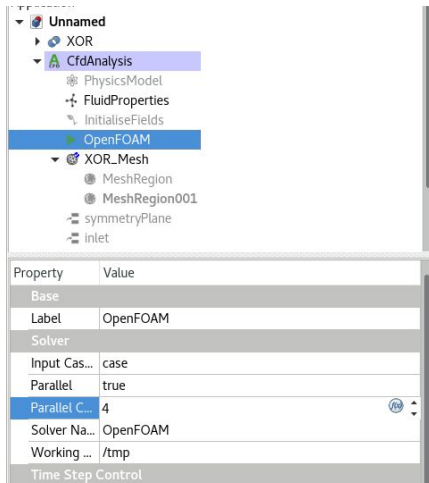
# Add boundary conditions

- Create a slip wall boundary condition for the outer face
- Create a 'Static pressure' outflow boundary and add the upper and back faces.
- The remaining settings include:
  - Fluid: air
  - Initialise with potential flow



# Run analysis

- For the 'OpenFOAM' solver object, set parallel processing to true and the desired number of parallel cores
- Double click on the 'OpenFOAM' object and click Write, then Run.

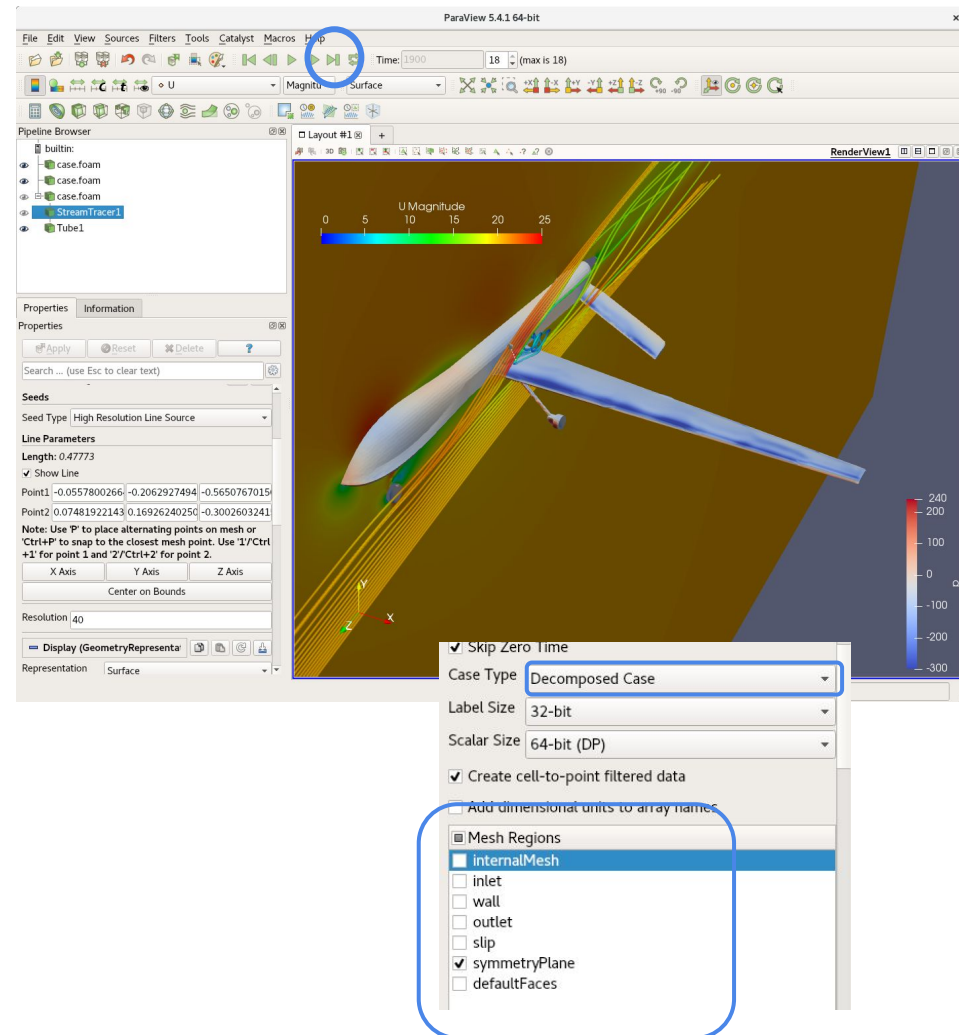


# Post-processing



# Visualisation

- Click 'Paraview', go to last time step
- Re-load 'pv.foam', select different patches to visualise if desired
- Add stream trace to object with internalMesh selected
- **Warning:** For incompressible, single phase solvers, OpenFOAM writes 'Kinematic pressure' =  $p/\rho$



# Integrated forces output

- While the analysis is running, click 'Edit' to open the case directory and edit the system/controlDict file.
- Paste the contents of the supplied 'forces' file at the end.
- Integrated forces and moments (about 'CofR') are written to the file:

```
postProcessing/forces_all/0/forces.dat
```

- Use to determine lift, drag, etc
- Re-run analysis with and without camera gimbal, gear retracted, etc, and find effect on forces

```
runTimeModifiable true;

libs
(
    // Needed for availability of porous baffle boundary in
    potentialFoam
    "libturbulenceModels.so"
);

functions
{
    forces_all
    {
        type            forces;
        libs             ( "libforces.so" );
        patches
        (
            wall
        );
        rho              rhoInf;
        rhoInf           1.2;
        log              off;
        writeControl      timeStep;
        writeInterval     1;
        CofR              ( 0 0 0 );
    }
}

// *****
//
```

The End